Contract # N00014-14-C-0020

Pilot-in-the-Loop CFD Method Development

Progress Report (CDRL A001)

Progress Report for Period: May 21, 2014 to June 30, 2014

PI: Joseph F. Horn
814-865-6434
joehorn@psu.edu

Performing Organization:

The Pennsylvania State University

Department of Aerospace Engineering

231C Hammond Building

University Park, PA 16802

Attn: Joseph F. Horn

Phone: 814-865-6434, Fax: 814-865-7092

Email: joehorn@psu.edu

Prepared under:

Contract Number N00014-14-C-0020
2012 Basic and Applied Research in Sea-Based Aviation
ONR #BAA12-SN-028
CDRL A001

Report Documentation Page

Form Approved
OMB No. 0704-0188

1. REPORT DATE **JUL 2014**	2. REPORT TYPE	3. DATES COVERED **21-05-2014 to 30-06-2014**
4. TITLE AND SUBTITLE **Pilot-in-the-Loop CFD Method Development**		5a. CONTRACT NUMBER 5b. GRANT NUMBER 5c. PROGRAM ELEMENT NUMBER
6. AUTHOR(S)		5d. PROJECT NUMBER 5e. TASK NUMBER 5f. WORK UNIT NUMBER
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) **The Pennsylvania State University,Department of Aerospace Engineering,231C Hammond Building,University Park,PA,16802**		8. PERFORMING ORGANIZATION REPORT NUMBER
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)		10. SPONSOR/MONITOR'S ACRONYM(S) 11. SPONSOR/MONITOR'S REPORT NUMBER(S)
12. DISTRIBUTION/AVAILABILITY STATEMENT **Approved for public release; distribution unlimited**		
13. SUPPLEMENTARY NOTES		
14. ABSTRACT		
15. SUBJECT TERMS		

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT **unclassified**	b. ABSTRACT **unclassified**	c. THIS PAGE **unclassified**	**Same as Report (SAR)**	**10**	

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std Z39-18

Report Distribution per CDRLs for Contract No. N00014-14-C-0020

ENCLOSURE NUMBER 1
CONTRACT DATA REQUIREMENTS LIST
INSTRUCTIONS FOR DISTRIBUTION

DISTRIBUTION OF TECHNICAL REPORTS, FINAL REPORT, THEORY AND USER MANUAL FOR SOFTWARE
(A SF 298 must accompany the final report)

		NUMBER OF COPIES	
ADDRESSEE	**DODAAC CODE**	**UNCLASSIFIED / UNLIMITED**	**UNCLASSIFIED/ LIMITED AND CLASSIFIED**
COR: Mr. John Kinzer ONR Code 351 E-Mail: john.kinzer@navy.mil	N00014	1	1
Program Officer: Dr. Judah Milgram ONR Code 351 E-Mail: judah.milgram@navy.mil	N00014	1	1
Program Officer: Ms. Susan Polsky NAVAIR 4.3.2.1 E-Mail: susan.polsky@navy.mil	N00024		
Administrative Contracting Officer* E-Mail: dcmaphiladelphiacasd@dcma.mil	S3915A	1	1
Director, Naval Research Lab Attn: Code 5596 4555 Overlook Avenue, SW Washington, D.C. 20375-5320 E-mail: reports@library.nrl.navy.mil	N00173	1	1
Defense Technical Information Center 8725 John J. Kingman Road STE 0944 Ft. Belvoir, VA 22060-6218 E-mail: tr@dtic.mil	HJ4701	1	1

* Send only a copy of the transmittal letter to the Administrative Contracting Officer; do not send actual reports to the Administrative Contracting Officer.

ELECTRONIC SUBMISSIONS OF TECHNICAL REPORTS IS PREFERRED AND ENCOURAGED. ELECTRONIC SUBMISSION SHOULD BE SENT TO THE E-MAIL ADDRESSES PROVIDED IN THE ABOVE TABLE, HOWEVER PLEASE NOTE THE FOLLOWING:

- Only Unlimited/Unclassified document copies may be submitted by e-mail.
- Unclassified/Limited has restricted distribution and a classified document (whether in its entirety or partially) is to be distributed in accordance with classified material handling procedures.
- Electronic submission to DIRECTOR, NAVAL RESEARCH LAB, shall be unclassified/unlimited reports and 30 pages or less. If unclassified and more than 30 pages,

Section I: Project Summary

1. Overview of Project

This project is performed under the Office of Naval Research program on Basic and Applied Research in Sea-Based Aviation (ONR BAA12-SN-0028). This project addresses the Sea Based Aviation (SBA) virtual dynamic interface (VDI) research topic area "Fast, high-fidelity physics-based simulation of coupled aerodynamics of moving ship and maneuvering rotorcraft". The work is a collaborative effort between Penn State, NAVAIR, and Combustion Research and Flow Technology (CRAFT Tech). This document presents progress at Penn State University.

All software supporting piloted simulations must run at real time speeds or faster. This requirement drives the number of equations that can be solved and in turn the fidelity of supporting physics based models. For real-time aircraft simulations, all aerodynamic related information for both the aircraft and the environment are incorporated into the simulation by way of lookup tables. This approach decouples the aerodynamics of the aircraft from the rest of its external environment. For example, ship airwake are calculated using CFD solutions without the presence of the helicopter main rotor. The gusts from the turbulent ship airwake are then re-played into the aircraft aerodynamic model via look-up tables. For up and away simulations, this approach works well. However, when an aircraft is flying very close to another body (i.e. a ship superstructure), aerodynamic coupling can exist. The main rotor of the helicopter distorts the flow around the ship possibly resulting significant differences in the disturbance on the helicopter. In such cases it is necessary to perform simultaneous calculations of both the Navier-Stokes equations and the aircraft equations of motion in order to achieve a high level of fidelity. This project will explore novel numerical modeling and computer hardware approaches with the goal of real time, fully coupled CFD for virtual dynamic interface modeling & simulation.

Penn State is supporting the project through integration of their GENHEL-PSU simulation model of a utility helicopter with CRAFT Tech's flow solvers. Penn State will provide their piloted simulation facility (the VLRCOE rotorcraft simulator) for preliminary demonstrations of pilot-in-the-loop simulations. Finally, Penn State will provide support for a final demonstration of the methods on the NAVAIR Manned Flight Simulator.

2. Activities this period

During the period of this report, the unsteady flow over the SFS2 ship model has been investigated using Craft Tech's flow solver CRUNCH CFD. An unstructured tetrahedral grid, generated by grid generation package Pointwise, has been provided by CRAFT Tech researchers. The unsteadiness of the flow was demonstrated by the velocity histories with respect to the time. Results have been validated with experimental data presented in [1].

CRUNCH-CFD is a multi-physics simulations tool for analyzing complex flow problems developed by CRAFT Tech. It has three different modules specialized on fluid and thermal problems: incompressible, compressible and thermal [2]. In the scope of this report, CRUNCH-CFD Incompressible module has been used for the unsteady airwake calculations of the SFS2 Ship model.

A 2500x2500x1000 ft. computational domain, shown in Figure 1, was generated by Pointwise mesh generation software. The ship body and sea surface were designated as viscous and inviscid wall boundaries, respectively. The subsonic inflow boundary applied to the inflow and far-field regions. The domain was chosen as large as possible to avoid the potential effects of grid size on the results. The full-scale (455x45x55 ft.) SFS2 (Simplified Frigate Shape 2) model is used. Since the size of the helicopter, which will be used in future studies, is relatively much smaller than the ship structure, a finer mesh topology should be created over and behind the ship deck to capture the detailed flow characteristics of ship airwake [1]. In order to that, the unstructured grid was clustered over the flight deck and behind the superstructure, which can be seen in Figure 2.

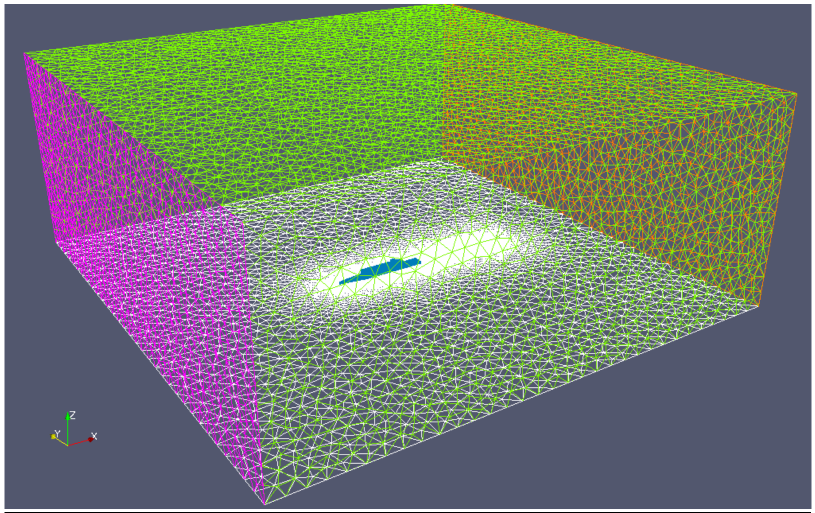

Figure 1 - Computational Domain

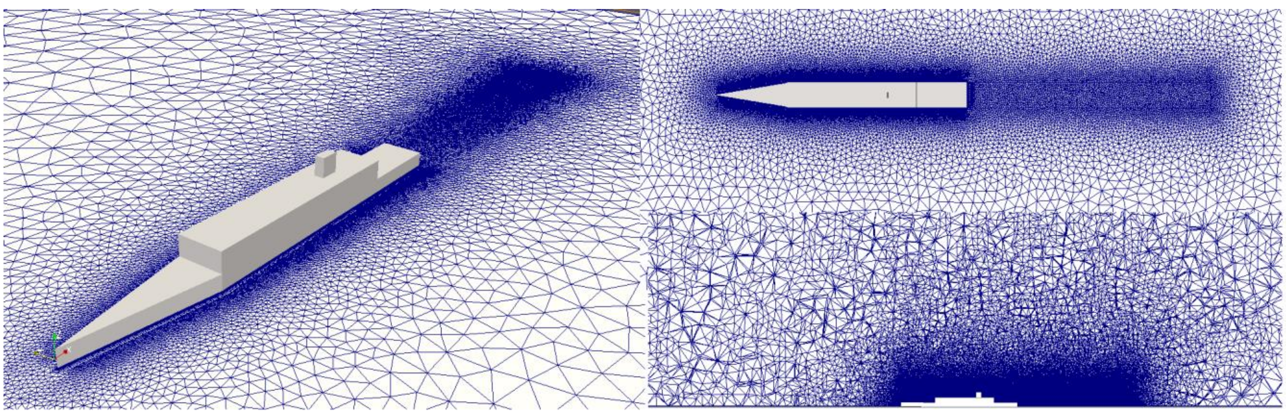

Figure 2 - Grid structure around SFS2 Ship model

The flow characteristics of the large ships are highly turbulent and unsteady. The superstructure of the ship produce large separated regions and vortex flows over and behind the ship. Because of these features of the flow, the airwake of the ship can only be captured with unsteady simulations. However, steady state simulations can capture the main tendency of the flow as averaged over a long period of time. And these results can be used as an initial condition for the unsteady CFD simulations to speed up the calculation process.

A steady state simulation with a Δt=0.001 sec time step was performed. The parallel computing was carried out using COCOA4 computer at Penn State with 32 processors. The solution convergence was determined by monitoring the history of the boundary flow residuals. Roughly, 2000 iterations were needed for the convergence.

The simulations were performed as unsteady laminar (MILES LES) which has been shown to be adequate for airwake simulations. Laminar flow assumption saves from time and computing power for the airwake simulations. The simulations were carried out with a free-stream velocity of V_∞ = 12 m/s (25 knots or 42.2 ft/s) and zero side-slip angle (β = 0 deg) .

Figure 3 and 4 shows the velocity magnitude distribution over the domain obtained from the steady state simulation. Three different flow separation regions can be observed from the results. The results show that steady state simulations can capture only the beginning of flow separation forming. Also it can be observed that the flow farther from the ship body is perfectly steady and the grid domain is big enough for the airwake region.

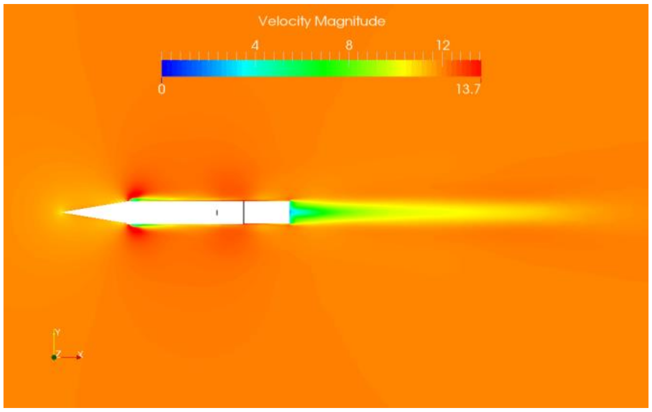

Figure 3 - Steady State CFD Case WODSpeed =12.87 m/s (23 Knots) WOD Angle = 0, XY plane

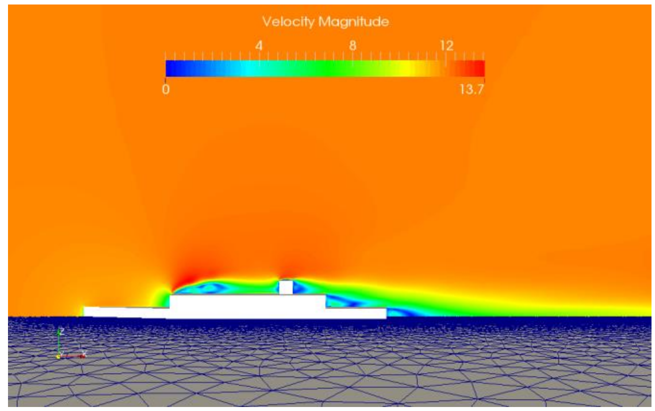

Figure 4 - Steady State CFD Case WODSpeed = 12.87 m/s (23 Knots) WOD Angle = 0, X-Z plane

After having acceptable results with steady state simulation, the simulation has been restarted using the steady state results as an initial condition for the unsteady CFD calculations. 1000 iterations (outer loop) with Δt=0.01 time step have been carried out to skip transition region for the turbulent flow development. After that, 6000 iteration with Δt = 0.01 has been carried out to capture ship airwake.

Figure 5 shows the velocity magnitude distribution over and behind the superstructure within a time period of 10 to 70 seconds of simulation. Shedding vortices from the superstructure can be observed in Figure 5. Results show that the periodicity of the shedding vortices was captured successfully by the flow solver.

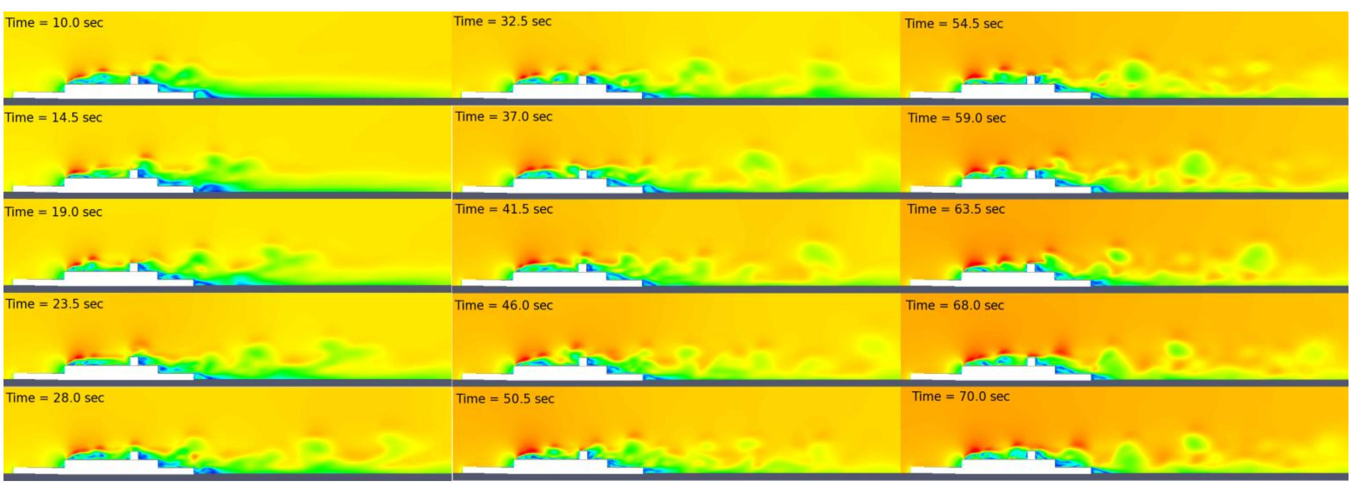

Figure 5 – Velocity magnitude distribution between 10th and 70th seconds of the simulation

The numerical results have been compared with the available corresponding time-averaged measurement data on the off-body planes in the flight deck and the superstructure regions, obtained from Ref. [1]. The experimental data is obtained from wind tunnel tests using a 1:8.5 SFS2 ship model.

Figure 6 shows the comparisons of the time averaged streamwise non-dimensional velocity distributions over flight deck between the measured experimental data and the computation. On Figure 6, the agreement is pretty good. Figure 7-8 show the quantitative comparison of streamwise and vertical velocity distributions over three different planes on the rear ship-deck, respectively. The plots show the measured data at 17 ft above the deck. The agreement is pretty good for the streamwise velocity component, but there is a small difference between calculated and measured vertical velocity distributions. This error might be a result of laminar flow assumption on numerical calculations. Also the experimental data belongs to a wind tunnel tests performed with a 1:8.5 SFS2 model, which can lead to scalability problems. Even though, the sideslip angle of the flow is zero and the ship geometry is symmetrical, there is some asymmetry in both computed and measured results, which is also observed by Zhang et al [1] and S.A. Polsky [3].

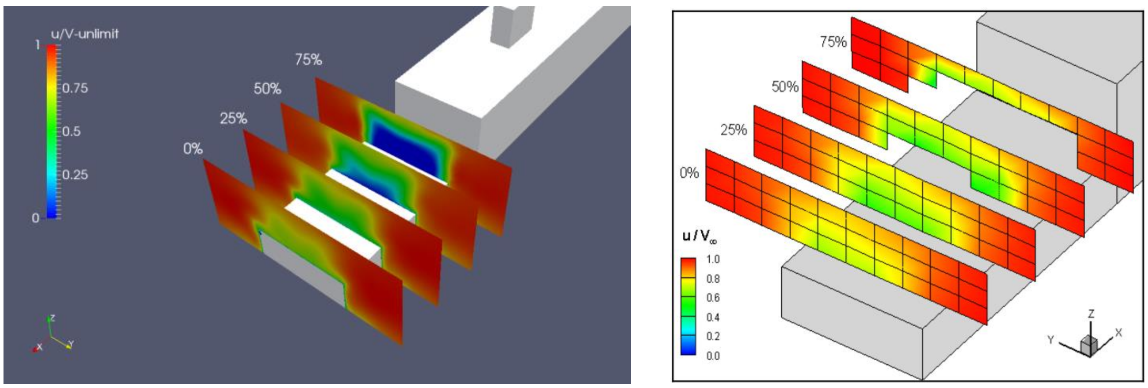

Figure 6 - a)Time-averaged streamwise velocity distributions over flight deck, CFD, b)Time-averaged stream wise velocity distributions over flight deck, experiment.[from 1]

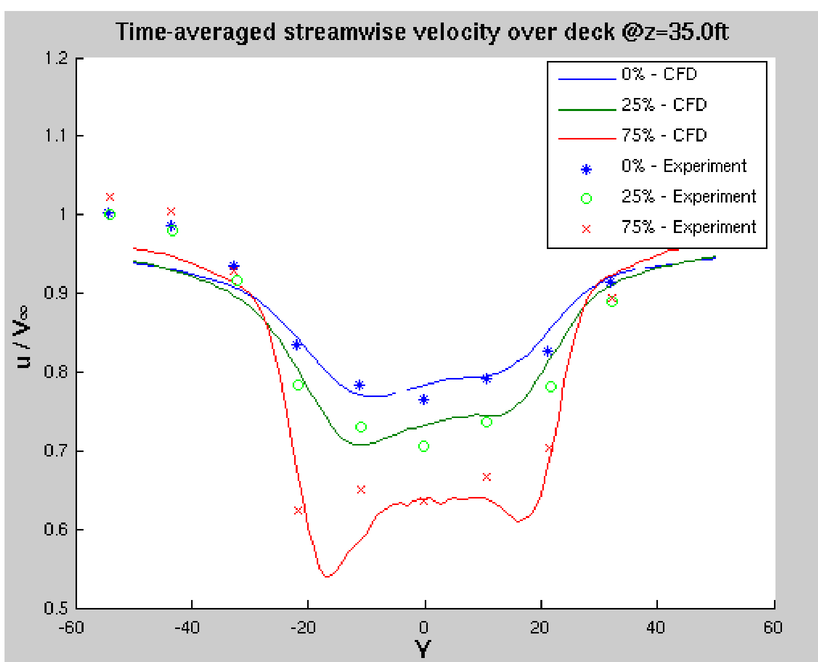

Figure 7 – Time averaged streamwise velocities over deck at different planes

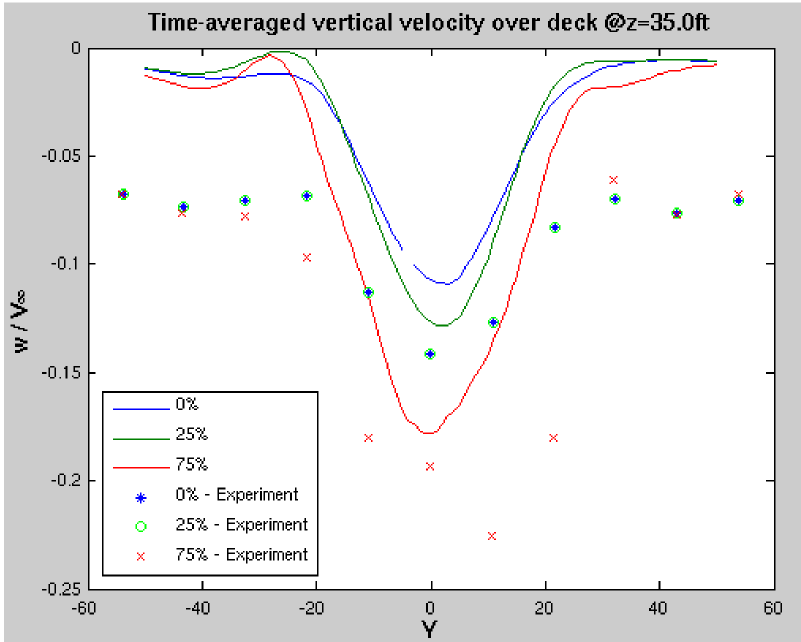

Figure 8 – Time averaged vertical velocities over deck at different planes

Figure 9 shows the comparison of time averaged streamwise velocity distributions between experimental data and calculated data over the super structure of the ship. The plots shows the measured data at 7.5 ft above super structure of the ship. The numerical results are in a good agreement with the measured data. The quantitative data is given in Figure 10 and 11. There are some offset between the calculated and the measured data. This offset might be because of the free-stream velocity difference between CFD and experiment. However the trend is correct and the numerical results are acceptable for the current phase of this research.

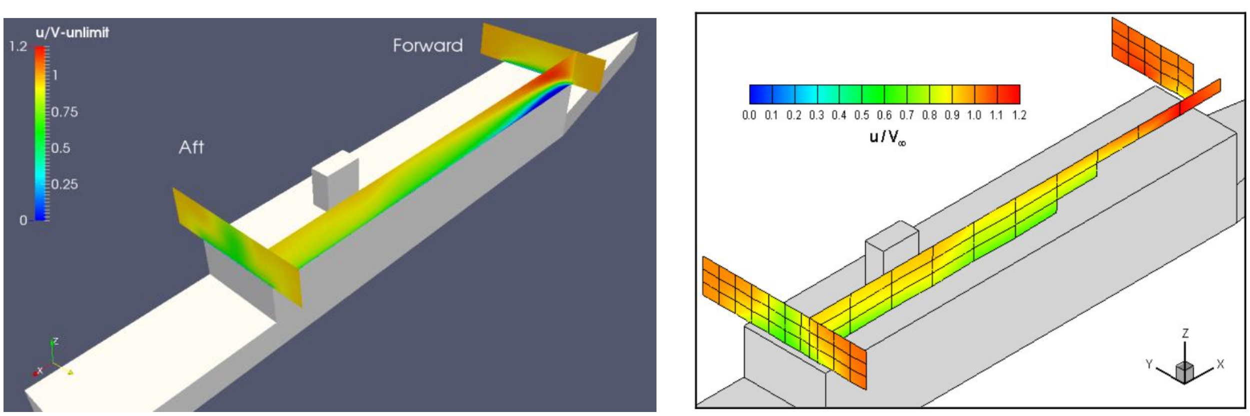

Figure 9 - a)Time-averaged streamwise velocity distributions over super structure, CFD, b)Time-averaged stream wise velocity distributions over super structure, experiment.[from 1]

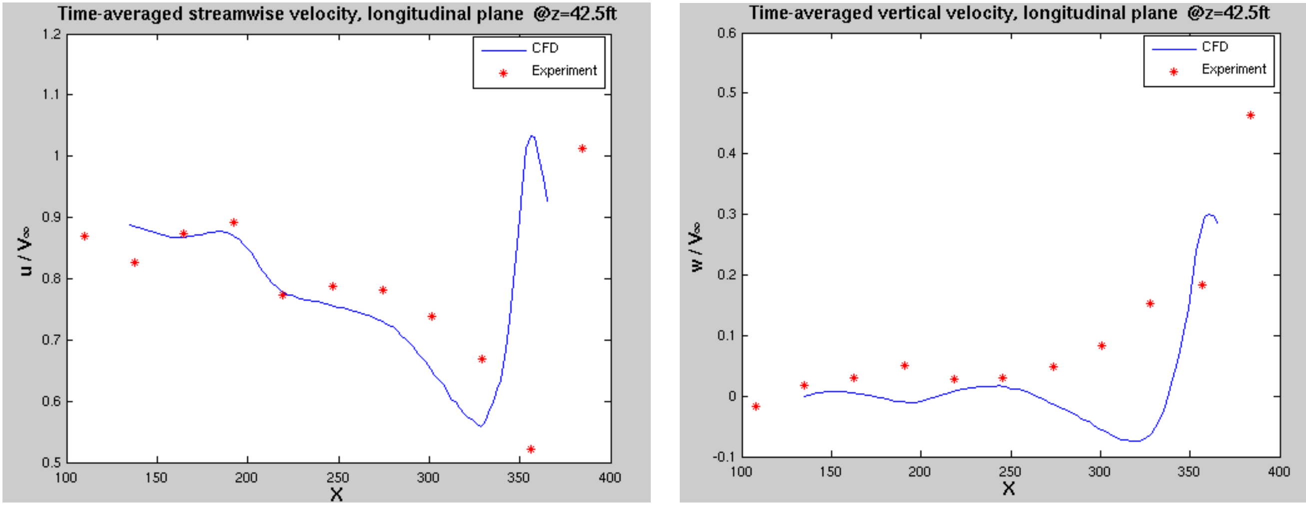

Figure 10 – a) Time averaged streamwise velocity, longitudinal plane (x positive toward ship bow) b)Time averaged vertical velocity, longitudinal plane

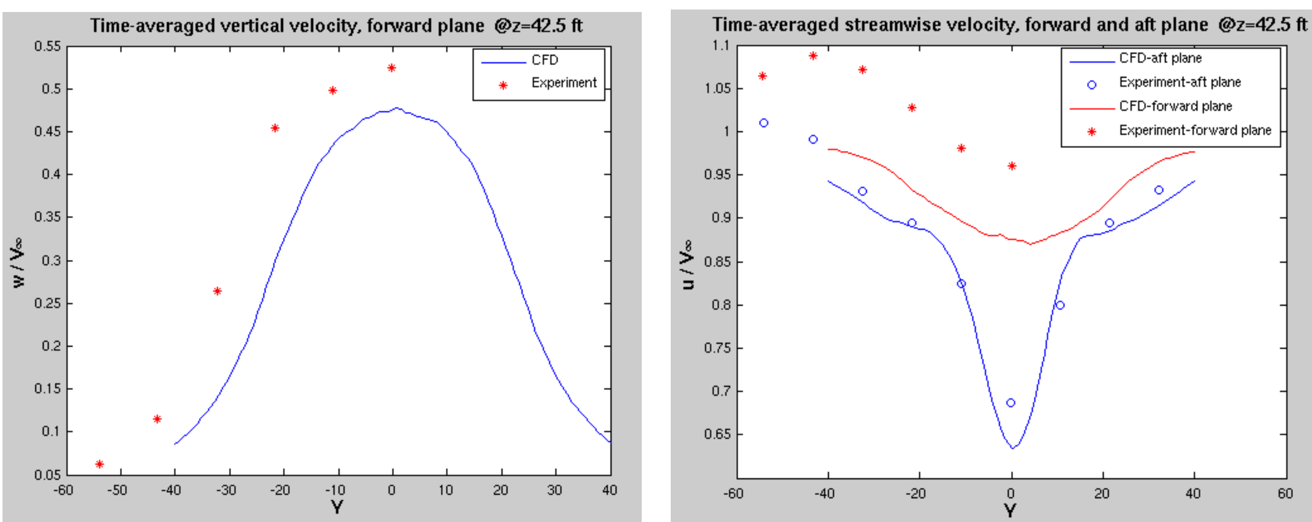

Figure 11 – a) Time averaged vertical velocity, forward plane b) Time averaged streamwise velocity, aft and forward plane

3. Significance of Results

The results obtained using CRUNCH CFD flow solver will be implemented to GENHEL-PSU in the next stage of this study.

4. Plans and upcoming events for next reporting period

- A subdomain region will be extracted from the full scale model.
- These flow solutions will be integrated with GENHEL-PSU to develop a set of baseline simulations of the utility helicopter operating in a ship airwake with one-way coupled flow solutions. This will provide a baseline with which to compare the fully coupled solutions.
- Begin development of fully-coupled simulations: In fully coupled solutions, blade position and aero loads are transmitted to the CFD code, the CFD code then calculates a velocity field (including the induced velocities from the aircraft airloads) and sends these velocity values back to the helicopter simulation model. The subsequent airloads and dynamics of the helicopter are then affected by the evolving external flow field. In this sense, the CFD solutions serve the purpose of not only the ship airwake effects but of the induced flow field generated by the helicopter main rotor (and possibly other components of the aircraft). Induced flow in the rotor is usually modeled by a lower order model in flight simulations (e.g. finite state inflow), but these modules will be replaced by CFD in the coupled solitions.
- Initial coupled solutions will not involve ship flow fields. Coupled simulations will be performed with the helicopter hovering in an open domain. The helicopter will be trimmed and perform an extended hover using the controller. The performance and trim of the helicopter will be compared to those predicted by the simulation model without coupled CFD. We expect to begin development of these solutions in June, with results expected later this summer.

5. References

1. Zhang F., and Xu. H., "Numerical Simulation of Unsteady Flow over SFS 2 Ship Model," AIAA-2009-0081, 47th Aerospace Sciences Meeting, Orlando, FL, Jan. 5-8, 2009.

2. Website, CRUNCH CFD by CRAFT Tech, https://crunch.craft-tech.com/products/crunch-cfd/, July, 11, 2014.

3. Polsky, S. A., Bruner, C. W., "A Computational Study of Unsteady Ship Airwake", NATO RTO AVT Symposium on "Advanced Flow Management: Part A – Vortex Flows and High Angle of Attack for Military Vehicles", Loen, Norway, May 2001.

6. Transitions/Impact

No major transition activities during the reporting period.

7. Collaborations

Penn State has collaborated with CRAFT Tech and conducted regular discussions with them.

8. Personnel supported

Principal investigator: Joseph F. Horn

Graduate Students: Ilker Oruc, PhD Student

9. Publications

No publications to date.

10. Point of Contact in Navy

Susan Polsky
Senior Computational Fluid Dynamics Specialist
Naval Air Systems Command Code 4.3.2.1
Applied Aerodynamics & Store Separation Branch
susan.polsky@navy.mil 301-342-8575 (Voice)

11. Acknowledgement/Disclaimer

This work was sponsored by the Office of Naval Research, ONR, under grant/contract number N00014-14-C-0020. The views and conclusions contained herein are those of the authors and should not be interpreted as necessarily representing the official policies or endorsements, either expressed or implied, of the Office of Naval Research, or the U.S. government.